SVHEAT™

2D / 3D Geothermal Modeling Software

Tutorial Manual

Written by:
Robert Thode, B.Sc.G.E., P.Eng.

Edited by:
Murray Fredlund, Ph.D., P.Eng.

SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada

Software License

The software described in this manual is furnished under a license agreement. The software may be used or copied only in accordance with the terms of the agreement.

Software Support

Support for the software is furnished under the terms of a support agreement.

Copyright

Information contained within this Tutorial Manual is copyrighted and all rights are reserved by SoilVision Systems Ltd. The SVHEAT software is a proprietary product and trade secret of SoilVision Systems. The Tutorial Manual may be reproduced or copied in whole or in part by the software licensee for use with running the software. The Tutorial Manual may not be reproduced or copied in any form or by any means for the purpose of selling the copies.

Disclaimer of Warranty

SoilVision Systems Ltd. reserves the right to make periodic modifications of this product without obligation to notify any person of such revision. SoilVision does not guarantee, warrant, or make any representation regarding the use of, or the results of, the programs in terms of correctness, accuracy, reliability, currentness, or otherwise; the user is expected to make the final evaluation in the context of his (her) own problems.

Trademarks

Windows™ is a registered trademark of Microsoft Corporation.
SoilVision® is a registered trademark of SoilVision Systems Ltd.
SVOFFICE ™ is a trademark of SoilVision Systems Ltd.
SVFLUX ™ is a trademark of SoilVision Systems Ltd.
CHEMFLUX ™ is a trademark of SoilVision Systems Ltd.
SVAIRFLOW ™ is a trademark of SoilVision Systems Ltd.
SVSOLID ™ is a trademark of SoilVision Systems Ltd.
SVHEAT ™ is a trademark of SoilVision Systems Ltd.
SVSLOPE ™ is a trademark of SoilVision Systems Ltd.
ACUMESH ™ is a trademark of SoilVision Systems Ltd.
FlexPDE® is a registered trademark of PDE Solutions Inc.

Copyright © 2008
by
SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada
ALL RIGHTS RESERVED
Printed in Canada

1 Introduction 4
2 Heated Pipeline 5
2.1 Model Setup 6
2.2 Results and Discussion 14
3 Heated Foundation 16
3.1 Model Setup 17
3.2 Results and Discussion 26
4 References 28
5 1D Simple Road pavement 30
5.1 Model Setup 31
5.2 Results and Discussion 46

1 Introduction

The Tutorial Manual serves a special role in guiding the first time users of the SVHEAT software through a typical example problem. The example is "typical" in the sense that it is not too rigorous on one hand and not too simple on the other hand.

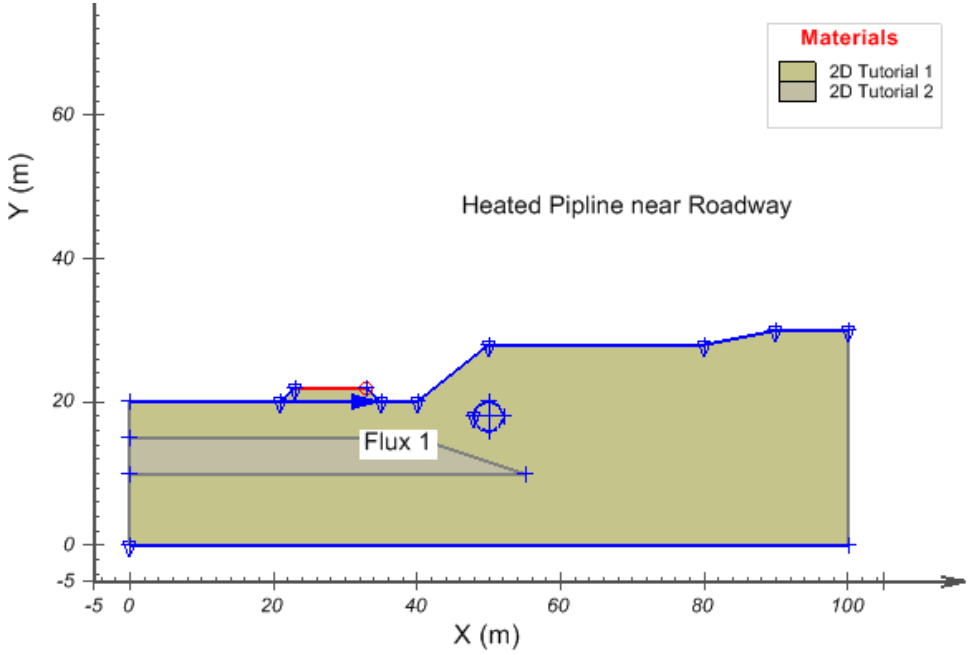
The Tutorial Manual serves as a guide by: assisting the user with the input of data necessary to solve the boundary value problem, ii.) explaining the relevance of the solution from an engineering standpoint, and iii.) assisting with the visualization of the computer output. An attempt has been made to ascertain and respond to questions most likely to be asked by first time users of SVHEAT.

2 Heated Pipeline

The following example introduces some of the basic features of SVHEAT and sets up a model of a buried pipeline. The purpose of this model is to determine the effects of the heated pipeline on the frozen ground and the nearby roadway. This steady-state model is composed of two regions and two materials. The model data and material properties are provided below.

Project: Roadway
Model: PipeLineTut2D
Minimum authorization required: STUDENT

Model Description and Geometry



Slope Region **Seam Region**

Shape 1 - polygon **Shape 2 -Circle**

X	Y			X	Y		X	Y
0	0		Center:	50	18		0	10
100	0						55	10
100	30		Radius:	2			40	15
90	30						0	15
80	28							
50	28							
40	20							
35	20							
33	22							
23	22							
21	20							

0	20							
0	15							
0	10							

Material Properties

The only soil properties required to solve this problem are those related to thermal conductivity since a steady-state analysis is being performed.

Material 1 - Slope Region: Thermal Conductivity curve laboratory data:

Temperature (ºC)	Conductivity (J/hr-m-ºC)
-10	1.58E+05
-1	1.57E+05
-0.1	1.56E+05
0	1.43E+05
0.1	1.28E+05
1	1.29E+05
10	1.30E+05

Material 2 - Seam Region: Thermal Conductivity curve laboratory data:

Temperature (ºC)	Conductivity (J/hr-m-ºC)
-10	2.00E+05
-1	1.90E+05
-0.1	1.80E+05
0	1.50E+05
0.1	1.30E+05
1	1.25E+05
10	1.20E+05

2.1 Model Setup

The following steps are required in order to set up and solve the model described in the preceding section. The steps fall under the general categories of:

a. Create model
b. Enter geometry
c. Specify boundary conditions
d. Apply material properties
e. Specify model output
f. Run model
g. Visualize results

a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Create a new project called "UserTutorial" by pressing the *New* button next to the list of projects,
3. Create a new model called "Heated_Pipeline" by pressing the *New* button next to the list of models. The new model will be automatically added under the recently created UserTutorial project. Use the settings shown below when creating a new model,
4. Select the following:

   Application: SVHEAT
   System: 2D
   Type: Steady-State
   Units: Metric
   Time Units: Seconds (s)
5. Click on the *World Coordinate System* tab by selecting the *World Coordinate System* tab on the *Create New Model* dialog,
6. Enter the World Coordinates System coordinates shown below into the dialog,

   x - minimum: -5
   y - minimum: -5
   x - maximum: 105
   y - maximum: 40
7. Click *OK* to close the dialog.

The workspace grid spacing needs to be set to aid in defining region shapes. The Seam Region in this example consists of a filter type material. The filter material of the model has coordinates of a precision of 0.5m. In order to draw any aspect of the geometry with this precision using the mouse, the grid spacing must be set to a maximum value of 0.5m.

1. On the Grid Spacing dialog enter 0.5 for both the horizontal and vertical spacing, and
2. Click *OK* to close the dialog.

b. Enter Geometry (Model > Geometry)

A region is the basic building block for a model. A region represents both a physical portion of material being modeled and a visualization area in the SVHEAT CAD workspace. A region will have a set of geometric shapes (e.g., polygons, circles, etc.) that define its material boundaries. Other modeling objects include features, such as flux sections, text, and line art. These features can be defined on any given region.

The present model is divided into two regions; namely the Slope region and the Seam region. Use the following steps to add the necessary regions:

1. Open the *Regions* dialog by selecting *Model* > *Geometry* > *Regions* from the menu,

2. Change the first region name from Region 1 (i.e., R1) to Slope. To do this, highlight the name and type the new name Slope,
3. Press the *New* button to add a second region,
4. Change the name of the second region to Seam, and
5. Click *OK* to close the dialog.

The shapes that define each material region are now created. Note that when drawing a geometric shape, information will be added to the region that is current in the Region Selector. The Region Selector is found at the top of the workspace and appears in the following image:

- **Define the Slope Region – Shape 1**
  Shape 1 refers to the entire region consisting of material 1.

1. Select "Slope" as the region by going to *Model* > *Geometry* > *Regions* and clicking on "Slope",
2. Select *Draw* > *Model Geometry* > *Region Polygon* from the menu,
3. The cursor will now be changed to cross hairs,
4. Move the cursor near (0,0) in the drawing space. You can view the coordinates of the current position of the mouse in the status bar just below the drawing space,
5. To select the point as part of the shape, left click on the point,
6. Now move the cursor near (100,0). Left click on the point. A line is now drawn from (0,0) to (100,0),
7. Repeat this process for the remaining points as provided at the beginning of this tutorial. This will define the geometry of the Slope Region,
8. For the last point (0,10), double-click on the point to finish the shape. A line is now drawn from (0,15) to (0,10) and the shape is automatically finished. SVHEAT will draw a line from (0,10) back to the starting point, (0,0).

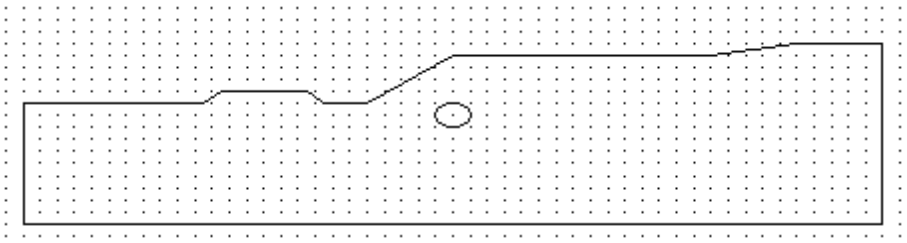
NOTE:
At times it may be somewhat tricky to snap onto a grid point that is near a line defined for a region. Turn the object snap off by clicking on *OSNAP* in the status bar to alleviate this problem. It should also be noted that shapes should, in this case, be drawn counter clockwise.

- **Define the Slope Region – Shape 2**
  Shape 2 refers to the geometry of the heated pipeline.

9. Ensure the Slope region is current in the region selector,
10. Select *Draw* > *Model Geometry* > *Region Circle* from the menu,
11. The cursor will now be changed to cross hairs,

12. Move the cursor near (50,18) in the drawing space. You can view the coordinates of the current position of the mouse in the status bar just below the drawing space,
13. To select the point as the circle center left click on the desired point,
14. Drag the cursor out to a *radius* of 2,
15. Let go of the mouse button to finish the circle,
16. To ensure that the radius is set to 2, open the double-click on the circle shape to open the *Region Properties* dialog,
17. Enter a radius of 2 if necessary,
18. Click *OK* to close the *Region Properties* dialog,

If the slope geometry been entered correctly the shape should look as follows:



NOTE:
If a mistake was made entering the coordinate points for a shape. Select a shape with the mouse and select *Edit* > *Delete* from the menu. This will remove the entire shape from the region. To edit the shape use the *Region Properties* dialog.

- **Define the Seam Region**
  The Seam Region corresponds to material 2 in the layout of the example,

19. Ensure that Seam is current in the region selector,
20. Select *Draw* > *Model Geometry* > *Region Polygon* from the menu,
21. The cursor will now be changed to cross hairs,
22. Move the cursor near (0,10) in the drawing space. You can view the coordinates of the current position of the mouse in the status bar just below the drawing space,
23. To select the point as part of the shape left click on the point,
24. Now move the cursor near (55,10). Left click on the point. A line is now drawn from (0,10) to (55,10),
25. Now move the cursor near (40,15). Left click on the point. A line is now drawn from (55,10) to (40,15), and
26. For the *last point* (0,15), left click to snap the cursor to the point. Double-click on the point to finish the shape. A line is now drawn from (40,15) to (0,15) and the shape of the seam is automatically finished. SVHEAT will draw a line from (0,15)

back to the start point, (0,10).

The diagram will appear as shown at the beginning of this tutorial after all the region geometries have been entered.

c. Specify Boundary Conditions (Model > Boundaries)

Boundary conditions must be applied to the points associated with each region. Once a boundary condition is applied to a boundary point, this defines the starting point for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until they are re-defined. The user may not define two different boundary conditions to the same line segment.

More information on boundary conditions can be found in *Menu System* > *Model Menu* > *Boundary Conditions* > *2D Boundary Conditions* in your User's Manual.

Now that all of the regions of the model geometry have been successfully defined, the next step is to specify the boundary conditions. An approximate geothermal gradient of 1°C/30m will be simulated by setting the temperature at the ground surface to -6 °C and the surface at the base of the model to -5 °C. The temperature of the pipe is 9 °C and the heat generated by the warming of the roadway is set as 200 kJ/hr (i.e., a thermal flux boundary condition). The steps associated with specifying the boundary conditions are as follows:

- **Slope Region – Shape 1**

1. Select the Slope dominant region in the drawing space,
2. From the menu select *Model* > *Boundaries* > *Boundary Conditions*. The boundary conditions dialog will open,
3. Select point (0,0) from the list,
4. From the *Boundary Condition* drop-down menu, select a "Temperature Expression" boundary condition. This will cause the Temperature Expression box to be enabled,
5. In the Constant/Expression box enter a temperature of -5,
6. Select the point (100,0) from the list,
7. From the Boundary Condition drop-down select a "Zero Flux" boundary condition,
8. Repeat for the remaining points referring to this table:

X	Y	Boundary Condition	Expression
0	0	Temperature Expression	-5
100	0	Zero Flux	
100	30	Temperature Expression	-6
90	30	Zero Flux	
80	28	Continue	
50	28	Continue	
40	20	Continue	
35	20	Continue	

33	22	Flux Expression	200000 (J/hr/m^2)
23	22	Temperature Expression	-6
21	20	Continue	
0	20	Zero Flux	
0	15	Continue	
0	10	Continue	

NOTE:
The Temperature Expression boundary condition for the point (100,30) becomes the boundary condition for the following line segments that have a Continue boundary condition until a new boundary condition is specified. By specifying a Flux Expression condition at point (33,22) the Continue boundary condition is stopped.

- **Slope Region – Shape 2**

9. Select the Slope circle region from the drawing space,
10. From the menu select *Model* > *Boundaries* > *Boundary Conditions* to open the *Boundaries* dialog,
11. In the Boundary Condition drop down select a Temperature Expression boundary condition,
12. Enter the value 9*,* and
13. Click *OK* to save the input Boundary Conditions and return to the workspace.

d. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the two materials that are used for the model. A material called *2D Tutorial 1* is defined for the major material region (i.e., Slope Region) and the material called *2D Tutorial 2* is defined for the smaller material region (i.e., Seam Region). This section will provide instructions on creating the first material. Repeat the process to add the other material.

1. Open the *Materials* dialog by selecting *Model* > *Materials* > *Manager* from the menu,
2. Click the *New...* button to create a material,
3. Enter 2D Tutorial 1 for the material name,
4. The Material Properties dialog should pop-up automatically,

NOTE:
When a new material is created, the display color of the material can be specified using the Fill Color box in the Material Properties menu. Any region that has a material assigned to it will display the corresponding material fill color.

5. Move to the *Conductivity* tab. Select *Data* from the Thermal Conductivity Option drop-down and press the *Data* button,
6. Enter the laboratory data points as provided in the "A Two Dimensional Example Model" section at the beginning of this Tutorial,

7. Click the Volumetric Heat Capacity tab,
8. Select the Constant radio button,
9. Enter 1.95e6 for the Frozen VHC and Unfrozen VHC text field,
10. Click *OK* on all open dialogs to accept the changes made and return to the *Materials Manager* menu, and
11. *Repeat* these steps to create the second material.

NOTE:
To view the thermal conductivity curve of the laboratory data press the *Graph* button on the bottom right of the *Thermal Conductivity Data* dialog.

The materials will also need to be applied to the model regions. Follow these instructions in order to assign the materials to regions:

1. Open the *Regions* dialog by selecting *Model* > *Geometry* > *Regions* from the menu,
2. For the Slope region, select *2D Tutorial 1* from the Material drop-down, and
3. For the Seam region, select *2D Tutorial 2* from the Material drop-down.

e. Specify Model Output

Flux sections can be used to show the rate of heat flow across a portion of the model for a steady state analysis and the rate of heat flow moving across a portion of the model in a transient analysis.

1. Select *Draw* > *Flux Section* from the menu to open the Flux Section,
2. Click on the point (21,20) with the mouse,
3. To finish the Flux Section, double-click on *point* (35,20). A blue line with an arrow on the end should be drawn across the geometry,
4. Default plots will be generated for reporting the results from the flux section will automatically be generated, and
5. Notice that the flux section label is partially on the region boundary in the workspace. To move the label location, select the textbox in the workspace and drag it to the desired location.

NOTE:
Flux Section labels can be formatted in the same manner as regular text boxes.

Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

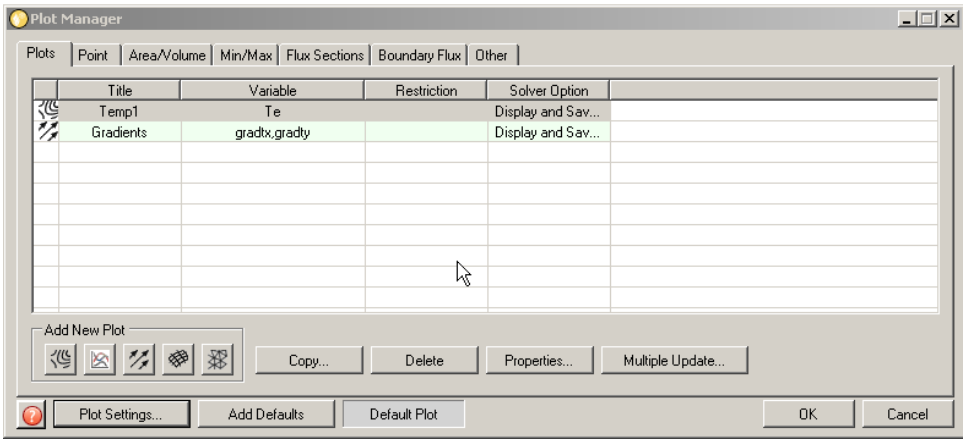
i) Plot Manager: Output displayed during model solution.
ii) Output Manager: Standard finite element files written out for visualization in

ACUMESH or for inputting to other finite element packages.

PLOT MANAGER **(Model > Reporting > Plot Manager)**

The *Plot Manager* dialog is first opened to display appropriate solver graphs. There are numerous plot types that can be specified to visualize the results of the model. Three plots will be generated for this tutorial example: temperature contours, thermal gradient vectors, and the final solution finite element mesh.

1. Open the *Plot Manager* dialog by selecting *Model* > *Reporting* > *Plot Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each plot type. Click on the *Contour* button to begin adding the first contour plot. The *Plot Properties* dialog will open,
3. Enter the title Temp1,
4. Select "Te" as the variable to plot from the drop-down,
5. Click *OK* to close the dialog and add the plot to the list,
6. Repeat these steps 2 to 5 to create the remaining plots shown below. Note that the Mesh plots do not require the entry of a variable,



Boundary Flux plots should also be created for this model. In order to do this, complete the following steps:

1. Select the *Boundary Flux* tab,
2. Add a New Report for the bottom of the Slope region. By default this boundary was named BN1 when the boundary condition was defined for this segment,
3. Select the Plot Off option for this Plot under the *Output options* tab,
4. Repeat steps 2 to 4 for the Circle shape's boundary (BN2 by default).
5. Click *OK* to close the *Plot Manager* and return to the workspace.

OUTPUT MANAGER **(Model > Reporting > Output Manager)**

Two output files will be generated for this tutorial example: a file of temperatures to be used as initial conditions in a subsequent transient analysis, and a .dat file to view the results in ACUMESH.

1. Open the *Output Manager* dialog by selecting *Model* > *Reporting* > *Output Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Create the temperature file by pressed the *SVHEAT File* button,
3. Click *OK* to close the dialog and add the output file to the list, and
4. Click OK to close the Output Manager and return to the workspace.

f. Run Model (Solve > Analyze)

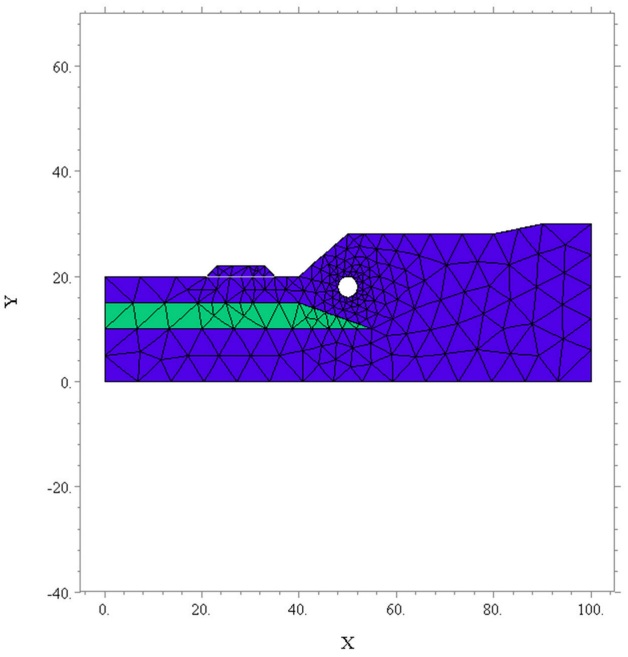
The next step is to analyze the model. Select *Solve* > *Analyze* from the menu. This action will write the descriptor file and open the FlexPDE solver. The solver will automatically begin solving the model.

g. Visualize Results (Window > AcuMesh)

The visual results for the current model may be examined by selecting the *Window* > *ACUMESH* menu option.

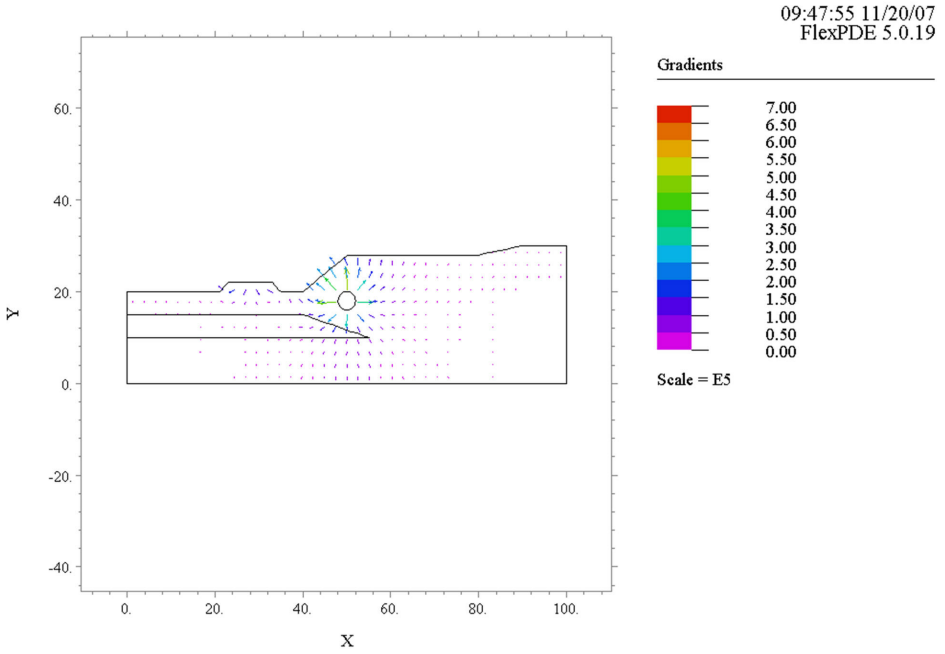
2.2 Results and Discussion

After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the SVHEAT solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.



The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas such as around the pipe contact where there is a significant change in temperature.

The temperature contours indicate that the temperature of the pipe does not significantly influence the base of roadway. Instead the heat flux due to the warming of the roadway surface has an influence of a few degrees.



Gradient Vectors show both the direction and the magnitude of the heat flow at specific points in the model. Vectors show the heat flow is away from the pipeline, as anticipated.

X Component of Flow in (J/hr)= 0.000000
Y Component of Flow in (J/hr)= -434461.1

Normal
1. On External Region Boundary: Positive Flow is into Region
2. On Internal Region Boundary - Region Given: Positive Flow is into Given Region
3. On Internal Region Boundary - Region Not Given: Positive Flow is into First Applicable Region
4. Internal: Left Hand Rule

Normal Flow in (J/hr)= 434461.1

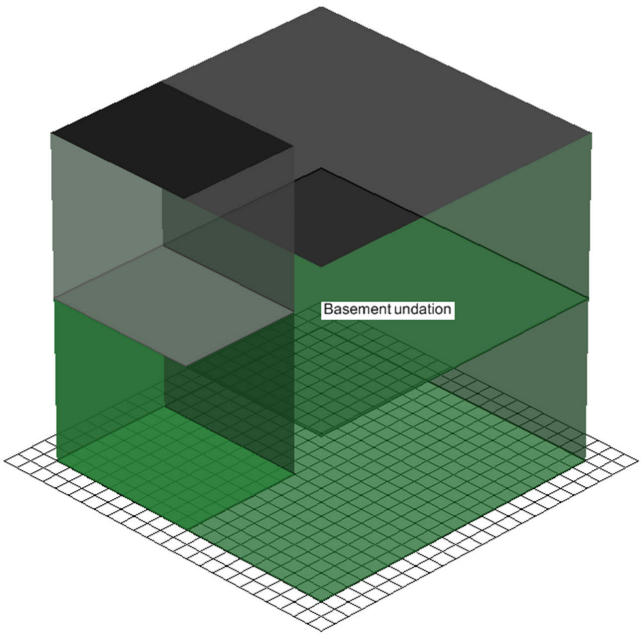
The Flux through the base of the roadway is displayed in the report dialog showing a breakdown of the *X, Y,* and normal components of flow through the model.

3 Heated Foundation

The following example will introduce you to a three-dimensional model in SVHEAT. The model is used to investigate steady-state heat flow through a material resulting from a heated foundation under winter conditions. The tutorial provides detailed set of instructions guiding the user through the creation of the 3D heat transfer model. The model is generated using two regions, three surfaces, and one material. The model data and material properties are provided below.

Project: Foundations
Model: HeatedFoundTut3D
Minimum authorization required: STUDENT

Model Description and Geometry



Region Shape Data for Tutorial – HeatedFoundation3D

Ground Region			Basement Region	
X	Y		X	Y
0	0		12	0
12	0		20	0
20	0		20	10
20	10		12	10
20	20			
12	20			
0	20			
0	10			

Material Properties

3D Tutorial Soil Material Properties

Temperature (°C)	Conductivity (J/s-m-°C)
-2	2000
-1	1999
0	1000
1	1001
2	1002

3.1 Model Setup

The following steps will be required in order to set up the model described in the preceding section. The steps fall under the general categories of:

a. Create model
b. Enter geometry
c. Specify boundary conditions
d. Apply material properties
e. Specify model output
f. Run model
g. Visualize results

a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Select the project called "UserTutorial" by pressing the *New* button next to the list of projects. If the project does not exist, create a new project called "UserTutorial",
3. Create a new model called "HeatedFoundation3D" by pressing the *New* button next to the list of models. The new model will be automatically added under the recently created Tutorial project. Use the settings shown below when creating the new model:

   Application: SVHEAT
   System: 3D
   Type: Steady-State
   Units: Metric
   Time Units: Seconds (s)

The *Settings* dialog will contain information about the current model System, Type, Units, and Time Units. The thermal conductivity data for the materials contained in the model are reported as J/hr-m-°C. In this case, the data for the model is in metric units and the units information will remain unchanged.

Before entering any model geometry it is best to format the Axes to ensure that the model will fit into the drawing space.

1. Select the *World Coordinate System* tab,
2. Enter the World Coordinates System coordinates shown below into the dialog.

   x - minimum: -10
   y - minimum: -10
   z - minimum: 0
   x - maximum: 30
   y - maximum: 30
   z - maximum: 20

3. Click the *OK* button to close the *Create New Model* dialog and to create the new model.

The workspace grid spacing needs to be set to aid in defining region shapes. The geometry data for this model has coordinates of a precision of 1m. In order to effectively draw geometry with this precision using the mouse, the grid spacing must be set to a maximum value of 1.

1. The *View Options* dialog will appear,
2. Enter *1* for both the horizontal and vertical spacing, and
3. Click *OK* to close the dialog.

b. Enter Geometry (Model > Geometry)

A region in SVHEAT is the basic building block for a model. A region represents both a physical portion of material being modeled and a visualization area in the SVHEAT CAD workspace. A region will have a set of geometric shapes that define its material boundaries. Also, other modeling objects including features, flux sections, text, and line art are defined on any given region.

This model will be divided into two regions, which are named Ground and Basement. Each region will have one of the materials defined as the region material. To add the necessary regions follow these steps:

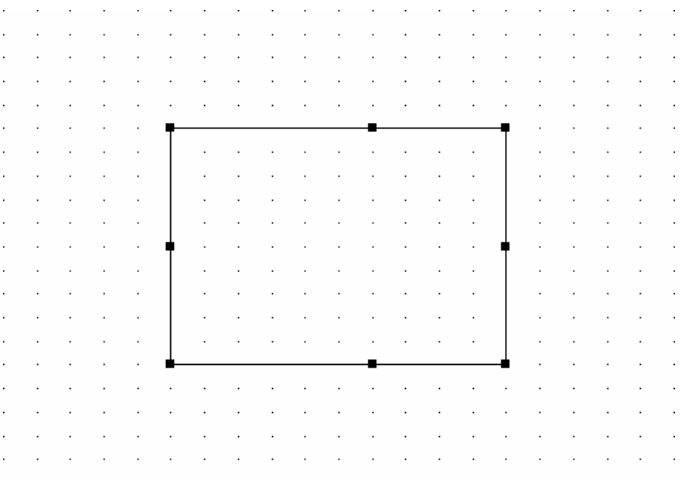
1. Open the *Regions* dialog by selecting *Model* > *Geometry* > *Regions* from the menu,
2. Change the first *region name* from "R1" to *Ground*. Highlight the name and type new text,
3. Press the *New* button to add a second region,
4. Change the name of the second region to Basement, and
5. Click *OK* to close the dialog.

The shapes that define each region will now be created. Note that when drawing geometry shapes the region that is current in the region selector is the region the geometry will be added to. The Region Selector is at the top of the workspace. Refer to the previous section of this manual for the geometry points for each region.

- **Define the Main region**

1. Specify "Ground" as the region by selecting *Model* > *Geometry* > *Regions* from the menu and clicking on "Ground",
2. Press *OK* to close the dialog,
3. Select *Draw* > *Model Geometry* > *Polygon Region* from the menu,
4. The cursor will now be changed to cross hairs,
5. Move the cursor near (0,0) in the drawing space. You can view the coordinates of the current position the mouse is at in the status bar just below the workspace,
6. To select the point as part of the shape left click on the *Point*,
7. Now move the cursor near (12,0) and then left click on the *Point*. A line is now drawn from (0,0) to (12,0),
8. Repeat this process for the remaining points. Refer to the previous section of this manual for the geometry points for each region,
9. For the last point (0,10). Double-click on the point to finish the shape. A line is now drawn from (0,20) to 0,10) and the shape is automatically finished by SVHEAT by drawing a line from (0,10) back to the start point, (0,0).

If the main geometry been entered correctly the shape should look like the following:



NOTE:
If a mistake was made entering the coordinate points for a shape then select a shape with the mouse and select *Edit* > *Delete* from the menu. This will remove the entire shape from the region. To edit the shape use the *Region Properties* dialog.

- **Define the Basement**

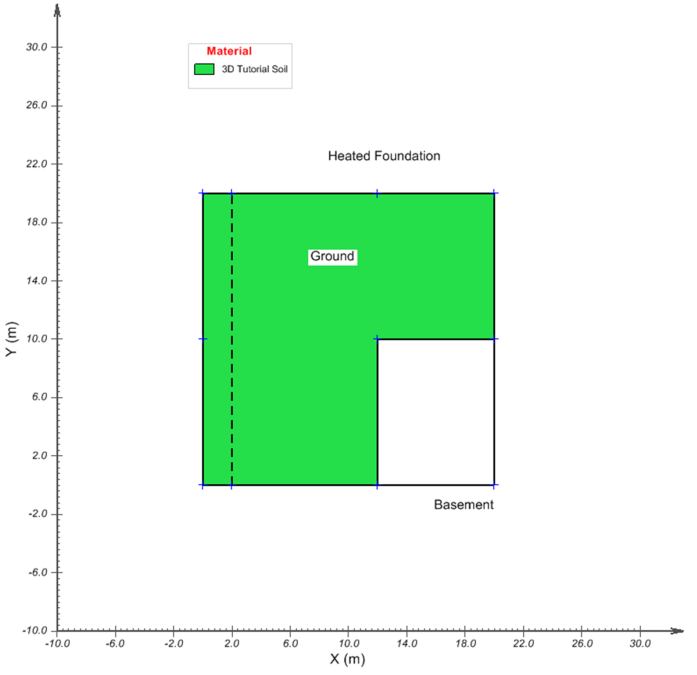
10. Select "Basement" as the region by going to *Model* > *Geometry* > *Regions*, and

clicking on *Basement*,

11. Press *OK* to close the dialog,
12. Select *Draw* > *Model Geometry* > *Region* *Polygon* from the menu,
13. Move the cursor near (12,0) in the drawing space. You can view the coordinates of the current position of the mouse in the status bar just below the drawing space,
14. To select the point as part of the shape left click on the *Point,*
15. Now move the cursor near (20,0) and then left click on the *Point*. A line is now drawn from (12,0) to (20,0),
16. Now move the cursor near (20,10) and then left click on the *Point*. A line is now drawn from (20,0) to (20,10), and
17. For the last point (12,10). Double-click on the *point* to *finish the shape*. A line is now drawn from (20,10) to 12,10) and the shape is automatically finished by SVHEAT by drawing a line from (12,10) back to the start point, (12,0).

NOTE:
At times it may be somewhat tricky to snap to a grid point that is near a line defined for a region. Turn the object snap off by clicking on "OSNAP" in the status bar to alleviate this problem.

After all the region geometry has been entered it will have the appearance of the diagram below (without the coloring).



This model consists of three surfaces. Although it is not required, the two surface grids have the same dimensions and grid densities. By default every model initially has two surfaces.

- **Define Surface 1**

  This surface is already present so the next step is to define the grid lines.

  1. Select "Surface 1" as the region by going to *Model* > *Geometry* > *Surfaces*, and clicking on *Surface 1*,
  2. Press *OK* to close the dialog.
  3. Click *Model* > *Geometry* > *Surface Properties* in the menu to open the *Surface Properties* dialog,
  4. Select "Elevation Data" from the Definition Options drop-down,
  5. Select the *Elevations* tab and click the *Define Grid lines...* button to set up the grid for the selected surface,
  6. There will be default grid lines of 0 and 10 present. Click the *Add Regular* button to open the *Add Regular X Gridlines* dialog,
  7. Enter -2 for Start, 1 for Increment Value, and 22 for End,
  8. Click *OK* to add the gridlines and *Close* the dialog,
  9. Click on *Define Grid* button again and move to the *Y* Grid Lines tab and repeat steps 7 to 8 for the *Y* gridlines,

Now that the grid has been set up, elevations must be specified for all the grid points:

  10. Enter *0* in the Set Nulls field,
  11. Click the *Set Nulls* button and all the missing elevations will be set to 0,
  12. Click *OK* to store the elevations and close the dialog,

- **Define Surface 2**

  This surface is already present. Follow the steps for defining Surface 1 elevations except enter 10 in the Set Nulls field and press the *Set Nulls* button. This will set the Surface 2 elevations to 10m.

- **Define Surface 3**

  The following steps are required to add the third surface to the model. Since Surface 2 and Surface 3 have the same grid lines, Surface 2 grid will be copied during insertion of the new surface:

  13. The *Surface* dialog can be opened by selecting *Model* > *Geometry* > *Surfaces* from the menu,
  14. Click "New" to open the *Insert Surfaces* dialog,
  15. Enter 1 as the Number of New Surfaces,
  16. Select to place the new surface *At The Top,*
  17. Select "Copy Grid" From An Existing Surface*,*
  18. Select "Surface 2" from the drop-down,

19. Choose to *Include* the elevations,
20. Enter 10m as the offset from Surface 2. This will set the elevation of the *Z*-axis to 20m,
21. Press *OK* to add the surface, and
22. Click *OK* to close the dialog.

c. Specify Boundary Conditions (Model > Boundaries)

Boundary conditions must be applied to region points. Once a boundary condition is applied to a boundary point this defines the starting point for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user may not define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in *Menu System > Model Menu > Boundary Conditions > 2D Boundary Conditions* in your User's Manual.

Now that all of the regions and surfaces have been successfully defined, the next step is to specify the boundary conditions on the region shapes. A temperature of −3°C will be defined on the Ground region to simulate an outdoor temperature. The basement will be set to a temperature of 10°C.

Boundary conditions can only be defined using a 2D view. The steps for specifying the boundary conditions are as follows:

1. Change to the 2D view by selecting *View > Mode > 2D*,
2. Specify Ground as the region by selecting *Model > Geometry > Regions* from the menu and clicking on "Ground",
3. Press *OK* to close the dialog,
4. From the menu select *Model > Boundaries > Boundary Conditions*. The *Boundary Conditions* dialog will open and display the boundary conditions for Surface 1. These boundary conditions will extend from Surface 1 to Surface 2 over Layer 1,
5. Select "Surface 3" from the drop-down box located at the top right of the dialog,
6. Under the *Surface Boundary Conditions* tab, select a Temperature Expression boundary condition from the surface boundary condition drop-down box,
7. Enter −3 in the Constant/Expression field. Press *OK* to close the dialog,
8. Specify Basement as the region by selecting *Model > Geometry > Regions* from the menu and clicking on Basement,
9. Press *OK* to close the dialog,
10. From the menu select *Model > Boundaries > Boundary Conditions*. The *Boundary Conditions* dialog will open to the *Segment Boundary Conditions* tab and display the boundary conditions for Surface 1,
11. Select "Surface 2" from the drop-down box,

12. Select the point (12,0) from the list,
13. From the Boundary Condition drop-down select a "No BC" boundary condition,
14. Select the point (20,10) from the list,
15. From the Boundary Condition drop-down select a "Temperature Expression" boundary condition from the surface boundary condition drop-down box. This will cause the Constant/Expression box to be enabled,
16. In the Constant/Expression box enter a temperature of 10, and
17. Press *OK* to close the dialog.

d. Apply Material Properties (Model > Materials)

The next step in defining the model is to enter the material properties for the material in the model. A clay is defined for both the ground and the basement. This section provides instructions on creating the clay material.

1. Open the *Materials* dialog by selecting *Model* > *Materials* > *Manager* from the menu,
2. Click the *New...* button to create a material,
3. Enter "3D Tutorial Soil" for the material name,
4. Press *OK.* The dialog for the new material properties will pop up,

NOTE:
When a new material is created, you can specify the display a color for the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned to it will display the material fill color.

5. On the *Conductivity* tab, click on the *Data* button beside the Unsaturated Thermal Conductivity (ensure the Data appears on the drop-down menu before clicking the *Data* button),
6. Enter the *laboratory data points* as provided in the "A Three Dimensional Example Model" section at the beginning of this tutorial,

NOTE:
To view the laboratory data press the *Graph* button.

7. Click the *Volumetric Heat Capacity* tab,
8. Select the *Constant radio* button,
9. Enter 1.95e6 for the Frozen VHC and Unfrozen VHC text field, and
10. Click *OK* on all opened dialogs to accept the changes and close the dialogs.

Each region will cut through all the layers in a model creating a separate "block" on each layer. Each block can be assigned a material or be left as *void*. A void area is essentially air space. In this model all "blocks" will be assigned a material.

1. Specify "Ground" as the region by selecting *Model* > *Geometry* > *Regions* from the menu and clicking on "Ground",
2. Press *OK* to close the dialog,
3. Select *Model* > *Materials* > *Material Layers* from the menu to open the *Material Layers* dialog,
4. Select the "3D Tutorial Soil" material from the drop-down for Layer 2,
5. Select the "3D Tutorial Soil" material from the drop-down for Layer 1,
6. Select "Basement" region by using the right arrow in the top right of the dialog,
7. Select "VOID" from the drop-down for *Layer 2,*
8. Select the "3D Tutorial Soil" material from the drop-down for *Layer 1,* and
9. Close the dialog using the *OK* button.

e. Specify Model Output

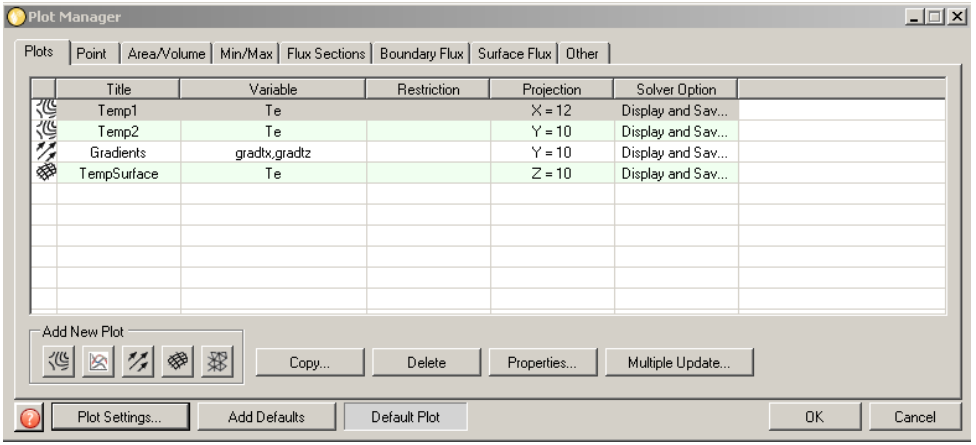
Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

i) Plot Manager:	Output displayed during model solution.
ii) Output Manager:	Standard finite element files written out for visualization in ACUMESH or for inputting to other finite element packages.

PLOT MANAGER **(Model > Reporting > Plot Manager)**

The *Plot Manager* dialog is first opened to display appropriate solver graphs. There are many plot types that can be specified to visualize the results of the model. A few plots will be generated for this tutorial example model including a plot of the temperature contours on both an *X* and *Y* plane, gradient vectors, temperature surface plot, and the solution mesh.

1. Open the *Plot Manager* dialog by selecting *Model* > *Reporting* > *Plot Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each plot type. Click on the *Contour* button to begin adding the first contour plot. The *Plot Properties* dialog will open,
3. Enter the title *Temp1,*
4. Select "Te" as the variable to plot from the drop-down,
5. Move to the *Projection* tab,
6. Select "Plane" as the Projection Option,
7. Select "*X*" from the Coordinate Direction drop-down,
8. Enter 12 in the Coordinate field. This will generate a 2D slice at X = 12m on which the temperature contours will be plotted,
9. Click *OK* to close the dialog and add the plot to the list,
10. Repeat steps 2 to 9 to create the remaining plots as shown below. Note that the Mesh plots do not require entry of a variable, and
11. Click *OK* to close the *Plot Manager* and return to the workspace.



OUTPUT FILES **(Model > Reporting > Output Manager)**

Two output files will be generated for this tutorial example model: a transfer file of temperatures, and a .dat file to transfer the results to ACUMESH.

1. Open the *Output Manager* dialog by selecting *Model* > *Reporting* > *Output Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each

output file type. Create the Te transfer file by pressing the *SVHeat* button,

3. Click *OK* to close the dialog and add the output file to the list, and
4. Click *OK* to close the *Output Manager* and return to the workspace.

f. Run Model (Solve > Analyze)

The next step is to analyze the model. Select *Solve* > *Analyze* from the menu. This action will write the descriptor file and open the SVHEAT solver. The solver will automatically begin solving the model.

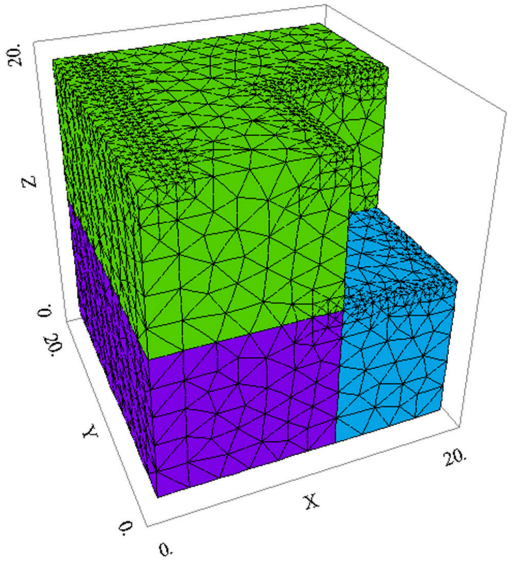
g. Visualize Results (Window > AcuMesh)

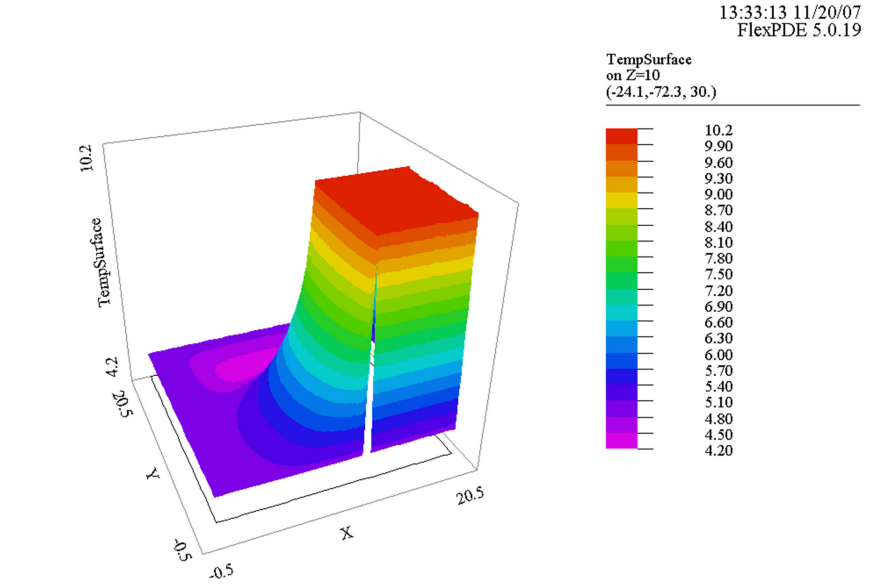
Once you have analyzed the model, the output plots can be visualized using ACUMESH. In order to view plots in ACUMESH, select *Window* > *ACUMESH* from the menu. Plots can visualized by selecting the desired process under Plots in the menu.

3.2 Results and Discussion

After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the SVHEAT solver.

Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots. This section will give a brief analysis for each plot that was generated.





Tutorial_HeatedFoundTut3D: Grid#4 p2 Nodes=25121 Cells=16396 RMS Err= 0.0059
Stage 1 Integral= 2541.165

4 References

FlexPDE 6.x Reference Manual, 2007. PDE Solutions Inc. Spokane Valley, WA 99206.

This page has been left intentionally.

5 1D Simple Road pavement

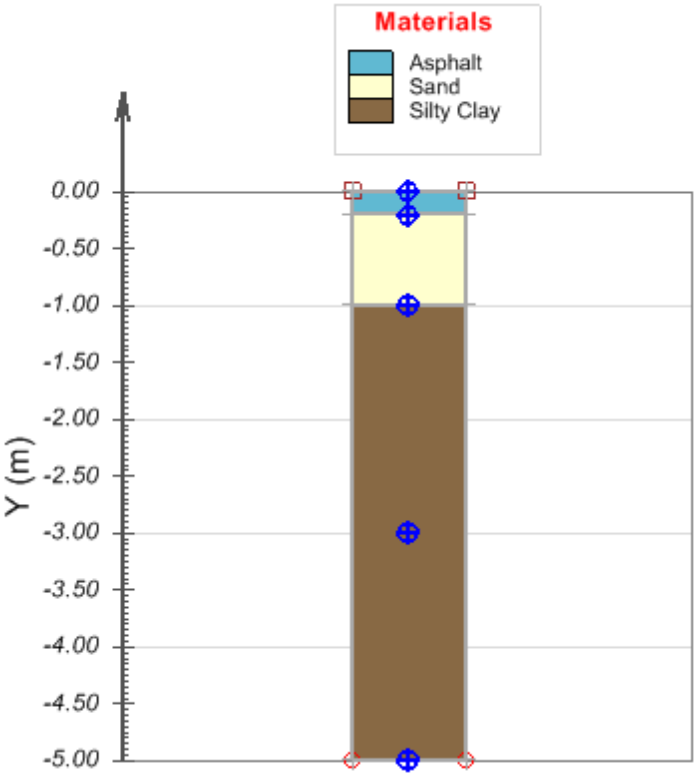
The purpose of this tutorial is to illustrate

- Procedures to create a 1D SVHeat model,
- Using different methods to specify thermal material properties,
- Applying the temperature as the climate boundary condition,
- Solve a transient problem of heat transfer.

Project: Roadway
Model: RoadPavement_TuT1D

Minimum authorization required: STUDENT

Model Description and Geometry



Region	Y coordinate
R1	0.0 to -0.2
R2	-0.2 to -1.0
R3	-1.0 to -5.0

5.1 Model Setup

The following steps are required in order to set up and solve the model described in the preceding section. The steps fall under the general categories of:

a. Create model
b. Create model geometry
c. Specify model global settings
d. Specify material properties
e. Apply material to the each region
f. Specify climate properties
g. Apply boundary conditions
h. Specify initial condition
i. Specify model output
j. Run model
k. Visualize results
l. Tune model settings

a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog. To Open SVOffice software, clicking Windows manual Start >> Programs >> SVOffice2009.
2. Select the project called "UserTutorial" by pressing the *New* button next to the list of projects. If the project does not exist, create a new project called "UserTutorial",
3. On the right side in "SVOffice 2009 Manager" dialog, click "New" button. After that, a dialog of "Create New Model" is opened. In this dialog, enter data or select the settings as shown following:

   In the **General** Tab the following settings must be specified.

   Application: Select "**SVHeat**" from the drop-down list,
   Model Name: Enter "**RoadPavement_TuT1D**" in the text box,
   System: Select "**1D Vertical**" from the drop-down list,
   Type: Select "**Transient**" from the drop-down list,
   Units: Select "**Metric**" from the drop-down list,
   Time Units: Select "**Day**" from the drop-down list.

   In the **World Coordinate System** Tab

   This tutorial is to simulate thermal transfer of soil below the road pavement. Enter Y coordinate as following:

	Minimum	Maximum
Y:	-5	0

   In the **Time** Tab

   The model is to simulate one year of climate temperature on the road pavement. Enter the end time of 365 days.

   Start Time: 0
   Initial Increment: 0.1
   Maximum Increment: 0.2

End Time: 365

NOTE: If the maximum increment time step is set too large, it may effect FEM solution convergence.

Double check the above model settings. If no problem, then Click **OK** button in the "Create New Model" dialog. After that, an "**Options**" dialog will be opened. Click "**OK**" button to close the dialog with the default settings.

b. Create Model Geometry

The model geometry is composed of 3 regions or layers, as shown in the following table:

Region Name	Y coordinate	Material
Region 1: R1	0 to -0.2 m	Asphalt
Region 2: R2	-0.2 to -1 m	Sand
Region 3: R3	-1 to -5 m	Silty Clay

Two methods can be used to create model geometry. One is to use menu of **Draw** >> **Model Geometry** >> **Region Polygon** to draw each region. Another method is to enter the coordinate for each regions. The coordinates for each region will be entered in this example.

The user must follow the steps below in order to enter the geometry for the correct result.

1. Click the menu of **Model** >> **Geometry** >> **Regions**. A dialog of "Regions" will be opened after clicking the menu.
2. Select the row with name of "**R1**" from the data grid table , and click " **Properties**..." button to edit the region properties.

NOTE: You can also open the **Regions Properties** dialog by double clicking the selected region in the data grid.

Region 1 (0 to -0.2 m)

3. In "**Region Properties**" dialog, click "**New Polygon...**" button,
4. Enter y coordinate of 0,
   After the value is entered, a new bank row will be added in the "New Polygon Shape" dialog,
5. Enter y coordinate -0.2 in the second row of data grid,
6. Click **OK** button,
7. Click **OK** button in "**Region Properties**" dialog.

Region 2 (-0.2 to -1 m)

8. Click **New** button in" **Regions"** dialog to create Region 2.
9. Select the row of **R2** from the data grid and then click the "**Properties...**" button.
10. In "**Region Properties**" dialog, click "**New Polygon...**" button
11. Enter y coordinate of -0.2
12. Enter y coordinate of -1 in the second row of data grid.
13. Click **OK** button

14. Click **OK** button in "**Region Properties**" dialog

Region 3 (-1 to -5 m)

15. Click **New** button in" **Regions"** dialog to create Region 3.
16. Select the row of **R3** from the data grid and then click the "**Properties...**" button.
17. In "**Region Properties**" dialog, click "**New Polygon...**" button
18. Enter y coordinate of -1
19. Enter y coordinate of -5
20. Click **OK** button
21. Click **OK** button in "**Region Properties**" dialog
22. Click **OK** button in "**Region**" dialog

c. Specify Model Global Settings (Model >> Settings)

SVHeat global settings allows user to specify different features such as thermal conduction, thermal convection, ice phase change, etc. Furthermore, soil water content can be specified in different approaches. Click the menu of **Model** >> **Settings...**. to open the dialog. However, this tutorial will use the default settings.

d. Specify Material Properties (Model >> Material)

Three materials of asphalt, sand, and clay, will be used in the model. The following steps are to specify the thermal material for each material.

Click the menu of **Model** >> **Material** >> **Manager.**

1. Click "**New**" button in the dialog of "**Material Manager**",
   Enter the name for a material
   Name: "asphalt",
2. Click "**OK**" button in the "New Material" dialog.

Specify thermal Conductivity for for asphalt material

3. Click "**Conductivity"** tab to specify thermal conductivity,
4. Select "**Constant**" from the drop-down list of "**Thermal Conductivity Option** ",
   Select the check box of "**Same value for unfrozen or frozen material**",
   Enter the constant value of thermal conductivity for asphalt
   Unfrozen Material: 103680 J/day-m-C,
   Thermal conductivity for asphalt material is estimated as 1.2 *w/m-C*, or 1.2 x 3600 x 24 = 103680 *J/day-m-C*.

Specify the Volumetric Heat Capacity for asphalt material

5. Click "**Volumetric Heat Capacity"** tab
6. Select "**Constant**" checkbox, and enter the value of heat capacity:
   Select the check box of "**Same value of unfrozen and frozen HC**",
   Enter the value of heat capacity
   Frozen Volumetric Heat Capacity: 2520000 J/m^3-C.

Specify Soil Freezing Characteristic curve (SFCC)

7. Click SFCC tab
   Select "**None**" from the drop-down list of "**SFCC method**", because the phase change for asphalt material is not needed to consider.

Specify the Volumetric Water Content (VWC)

8. Click "**VWC**" tab,
   The water content of asphalt can be neglected, Enter a small amount of water content,
   SatVWC: 0.001
   VWC: 0.0001
9. Click **OK** button.

Specify Thermal properties for sand material

10. Click "**New**" button in the "Material Manager" dialog,
    Enter material name: sand,
11. Click "**OK**" button in the "New Material" dialog

Specify Thermal Conductivity for for **sand** material

12. Click "**Conductivity"** tab,
    Select "**Johansen**" approach from the drop-down list,0
    With the Johansen approach, the thermal conductivity is calculated based on water content, ice content and soil solid component fraction.
    Material State: Crushed,
    Material Type: Fine,
    Solid Conductivity: checked,
    Solid Component: 734400 *J/day-m-C*,
    The estimated value of thermal conductivity solid Component = 734400 *J/day-m-C*.

Specify Volumetric Heat Capacity for asphalt material

13. Click "**Volumetric Heat Capacity"** tab,
    Select the radio button of "**Jame-Newman**",
    Solid Dry Density: 1600 kg/m^3,
    Specific Heat Capacity of Solid Component: 700 *J/kg-C*,

Specify Soil Freezing Characteristic curve (SFCC)

14. Click SFCC tab
    Enter the interval of phase change temperature for sand,
    From: -0.01 C,
    To: -0.5 C

    Select the "Estimated By SWCC" from drop-down list.

    It is very important to specify the relationship between of unfrozen water content and the temperature. If data of unfrozen water content is unavailable, it can be estimated with soil water characteristic curve (SWCC).

    Select *Fredlund and Xing Fit* from the drop-down list of SWCC Methods,
    Click SWCC Properties... button,
    Enter the following value for the SWCC fitting parameter.

    af: 29.4 *kPa*

nf: 6.04
mf: 0.335
hr: 105.9 *kPa*

Click Graph SWCC... button to preview SWCC as shown in Figure 1.

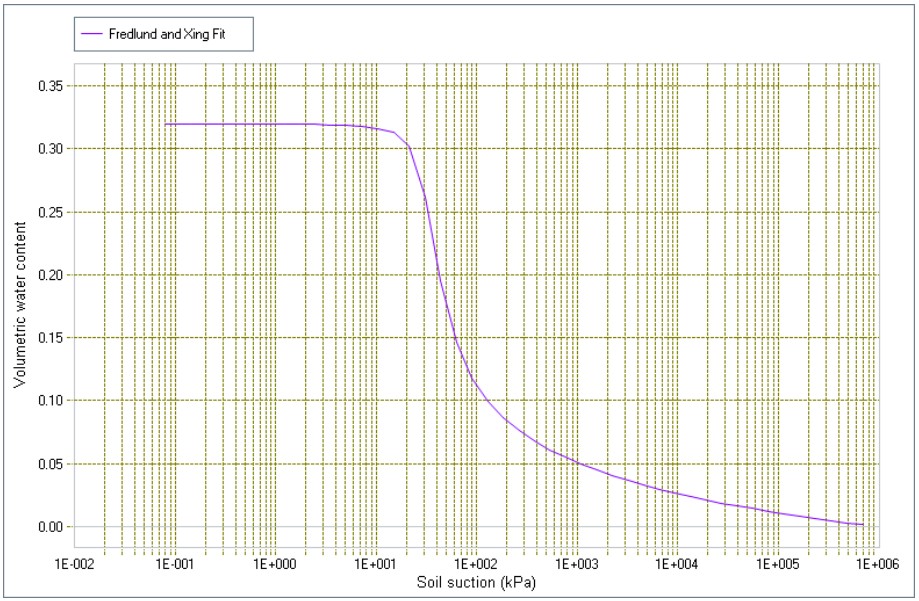


Figure SWCC that is used to estimate SFCC

Specify Volumetric Water Content (VWC) of sand material

15. Click "**VWC**" tab,
    Enter a constant value of sand water content,
    SatVWC: 0.32
    VWC: 0.2
16. Click "**OK**" button.

Specify thermal properties for clay material

17. Click "**New**" button in the "Material Manager" dialog,
    Enter material name: silty clay,
18. Click "**OK**" button in the "New Material" dialog

Specify Thermal Conductivity for for sand material

19. Select "DeVries" approach from the drop-down list,
    Enter the thermal Conductivity of each phase in soil.

    Solid Phase: 708480 *J/day-m-C*
    Liquid Phase: 52272 *J/day-m-C*
    Dry air Phase: 2073 *J/day-m-C*
    Vapor air: 466560 *kg/m³*

Specify Volumetric Heat Capacity for asphalt material

20. Click "**Volumetric Heat Capacity"** tab,
21. Select radio button of "**Jame-Newman**",
    Solid Dry Density: 1320 kg/m^3
    Specific Heat Capacity of Solid Component: 800 *J/kg-C*

Specify Soil Freezing Characteristic curve (SFCC)

22. Click SFCC tab
    Enter the interval of phase change temperature for sand,
    From: -0.05 C,
    To: -1.50 C
23. Select the "Tice & Anderson Fit" from "**SFCC Method**" drop-down list.
    Enter the value for parameter A: 0.035
    parameter B: 0.450
    Soil dry density: 1320 *kg/m³*
24. Click "**OK"** button.

Specify Volumetric Water Content (VWC) of clay material

25. Click "**VWC**" tab,
    Enter a constant water content of clay material,
    SatVWC: 0.45 *m³/m³*
    VWC: 0.3 *m³/m³*

26. Click "**OK**" button in the "**Material Properties**" dialog.
27. Click "**OK**" button in the "**Material Manager**" dialog.

e. Apply Material to Each Region

1. Click menu of **Mode**l >> **Geometry** >> **Regions**,
2. Select the **R1** region in the data grid,
3. Click "**Properties**" button,
4. Select "**asphalt**" from the "**Material**:" drop-down list on the bottom-left corner in the dialog,
5. Click "**OK**" button

6. Select "**R2**" region in the data grid,
7. Click "**Properties**" button,
8. Select "**sand**" from the "**Material**:" drop-down list on the bottom-left corner in the dialog,
9. Click "**OK**" button.

10. Select "**R3**" region in the data grid,
11. Click "**Properties**" button,
12. Select "**silty clay**" from the "**Material**:" drop-down list on the bottom-left corner in the dialog,
13. Click "**OK**" button in "**Region Properties**",
14. Click "**OK**" button in "**Regions**" dialog.

NOTE: You can open a particular region properties dialog by moving mouse

onto a particular region, and then double clicking the region, as shown in Figure 2.

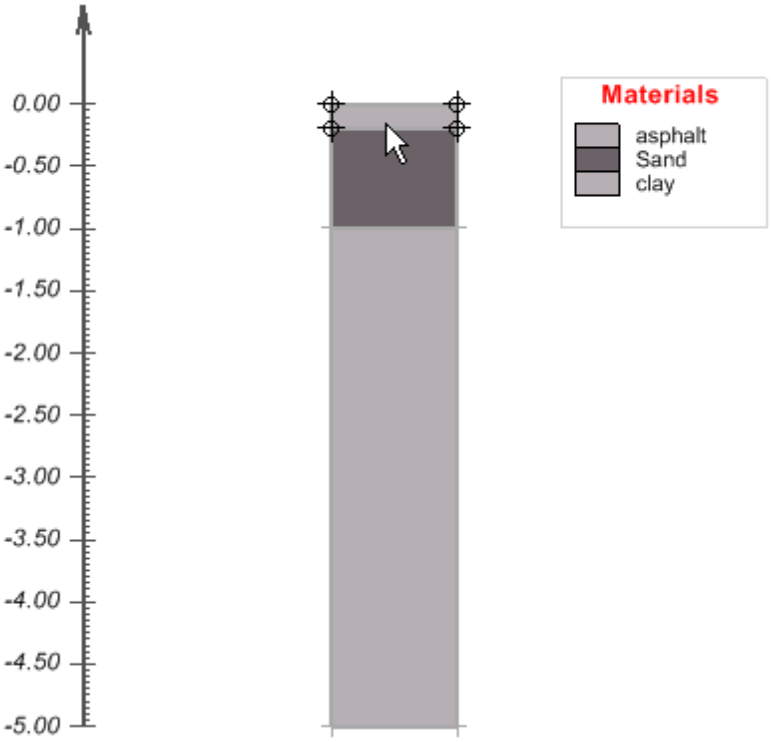


Figure 2 Fast open a region properties dialog

f. Specify Climate Propeties (Model >> Boundaries >> Climate Manager)

To simulate climate temperature effect on heat treanser, use the climate manager to specify climate properties.

1. Click the menu of Model >> Boundaries >> Climate Manager,
2. Click "New" button in the "Climate Manager" dialog,
3. Enter a climate name in the text box,
   Name: DemoClimate
4. Click "OK" button,

Specify an approach to determine the temperature at the ground surface

Generally, the temperature at the ground surface is different from the air temperature. In this tutorial, N-Factor approach is used.

5. Click "General" Tab
6. Select "Empirical with N-Factor" from the drop-down list,
7. Click "N-Factor" tab,
8. Select "Constant" from the drop-down list,
9. Enter an estimated value of N-Factor,
   N-Factor Constant: 0.8
10. Click "Air Temperature" Tab,

11. Select "Constant/Expression" from the drop-down list,
12. Enter the following formula to simulate daily changing temperature, shown in Figure 3.
    Constant/Expression: *1.2 + 15 * sin(2*3.141596/365*t + 3.141596/2)*
    where:
    t = time. It is a valid variable in PDE script.

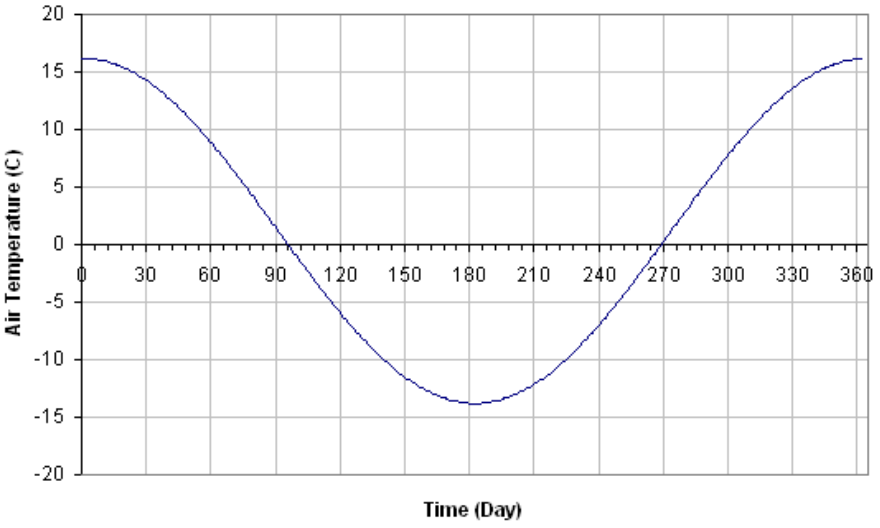


Figure 3 Air temperature changing with time

13. Click "**OK**" button in "**Climate Properties**" dialog,
14. Click "**OK**" button in "**Climate Manager**" dialog,

g. Apply Boundary Conditions (Model >> Boundaries >> Boundary Conditions)

Apply the climate to the ground surface.

1. Select **R1** geometry by moving mouse to and clicking the region, as shown in Figure 2,
2. Click the menu of **Model** >> **Boundaries** >> **Boundary Conditions**,
3. Select the (first) row with Y coordinate of 0.00 from the data grid in "**Boundary Condition**" dialog,
4. Select "**Climate**" from the "**Boundary Condition:**" drop-down list,
5. Select "DemoClimate" from the "**Climate Name**:" drop-down list, as shown in Figure 4.
6. Click "**OK**" button.

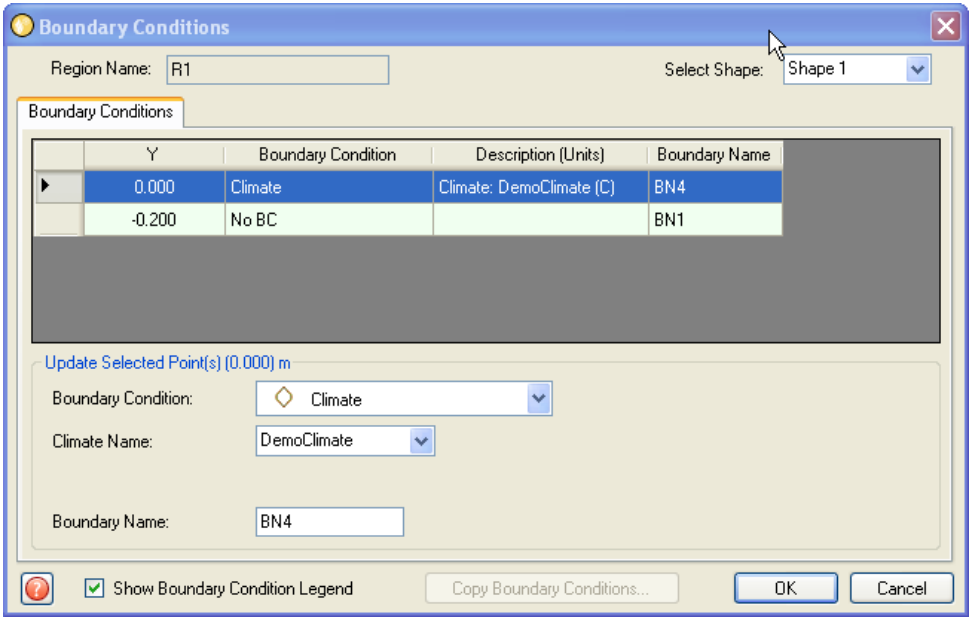


Figure 4 Apply climate boundary on the ground surface

Apply thermal flux at the bottom of model geometry

7. Select **R3** by moving mouse to the region 3 and clicking the region,
8. Click the menu of **Model** >> **Boundaries** >> **Boundary Conditions**,
9. Select the (2nd) row with Y coordinate of -5.00 from the data grid in "Boundary Conditions" dialog,
10. Select "**Flux Expression**" from the "**Boundary Conditions**:" drop-down list,
11. Enter a value in "**Constant/Expression:** " text box,
    Constant/Expression: 5148 *J/day-m²*

    The estimated thermal flux on the bottom is 0.06 *w/m²* = 5148 *J/day-m²*.

12. Click "**OK**" button.

h. Specify Initial Conditions (Model >> Initial Conditions >> Settings)

The transient problem must specify the initial conditions.

1. Click the menu of **Model** >> **Initial Conditions** >> **Settings**...
2. Select "**Constant/Expression**" radio button,
3. Enter the initial temperature: 13 *°C*
4. Click "**OK**" button.

i. Specify Output Settings (Model >> Reporting >> Plot Manager)

Specify output settings for analysis of model running results.

1. Click the menu of **Model** >> **Reporting** >> **Plot Manager**,
2. Click "**Plots**" Tab

In Plots tab, some default outputs are created. You can modify the output settings for your purpose.

3. Select the row with the title of "**Temp**",
4. Click "**Properties**..." button, or double clicking the selected row.
5. Click "**Description**" tab,
6. Change "**Temp**" into "MyTemp" in the "Title" text box,
7. Click "**Update Method**" tab,
8. Change increment as
   Increment: 1
9. Click "Output Options" Tab,
10. Check "**Write.txt File**" check box
    A text file with file name of MyTemp.txt will be created under the directory of model file after running the model.
11. Click "**OK**" button.

Specify temperature output at a particular location

12. Click "Point" Tab,
13. Click "**Multiple Entry**" button,
14. Select the "Te" variable from the drop-down list,
15. Enter a group name in the text box,
    "**Group**:" TempAtEachLayer
16. In the data grid table, enter Y coordinate and tile as following,

0	temp0
-0.2	temp02
-0.5	temp05
-1.0	temp1
-2.0	temp2
-3	temp3
-5	temp5

17. Click "**OK**" button,
18. Select all rows in "**Point**" tab,
19. Click "**Multiple Update...**" button,
20. Set increment step to 1
    Increment: 1
21. Click "**Output Options**" tab,
22. Select the check box of Write to .txt File, if not checked,
    A file TempAtEachLayer.txt will be created under the directory of model file after running the model.
23. Click "**OK**" button
24. Click "**OK**" button in "**Plot Manager**" dialog

Now the model settings are completed (Figure 5).

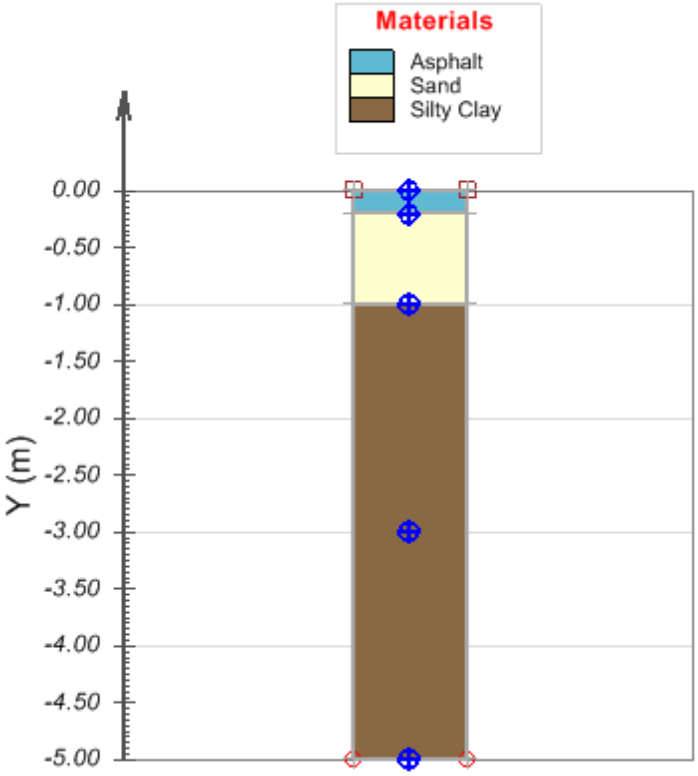


Figure 5 Model geometry and settings.
The points are the location to display temperature changing

j. Run Model (Solve >> Analyze)

To run the model, click menu of **Solve** >> **Analyze**.

k. Analysis of Result

To analyze the running results, one approach is to use AcuMesh tools of SVOffice, Another is to use the output data file that was created under the directory of the model file.

l. Tuning model settings

If the result is not expected, try to tune the material properties. Set a reasonable value for thermal conductivity, heat capacity, and unfrozen water content as the function of temperature, etc. Also it is very important to specify correct boundary conditions to obtain a proper solution. After a model setting is changed, it is recommended to save the model as a separate name.

Change climate temperature based on daily minimum and maximum temperature

In Figure 3, the yearly climate temperature change is approximated with a empirical formula. However, the daily climate data are recorded in a weather station. SVHeat allows user to specify the temperature based on daily average temperature or daily minimum and maximum temperature. The following is to modify model settings to use the daily min/max

temperature changing pattern:

1. Save model as a separate model name,
Click the menu of **File >> Save As**. A dialog will be opened. In this dialog, do the following:

In General Tab,
Enter the "**New File Name"**: SimpleRoadPavement_1D_MinMaxTemp

Click **OK** button

2. Modified climate temperature settings
Click the menu of **Model >> Boundaries >> Climate Manager** … ,
Select the row of "**DemoClimate**" from the data grid,

Double click the (2nd) column of "Applied" in the selected row to open the "Climate Properties" dialog.
Click "**Air Temperature**" Tab
Select "**Data – Smooth Cos Function**" from the "**Air Temperature Option**:" drop-down list.

Enter the following time and min/max temperature into the data grid. Here only 5 daily dataset is used for example.

Time (day)	Min Temp (C)	Max Temp (C)
0	15.6	16.2
1	14.3	27.6
2	16.2	18.7
3	14.5	16.5
4	14.8	17.0

NOTE:
For a large dataset, you can copy dataset from Microsoft Excel and then paste to the dataset of the data grid in the dialog.

Enter the time that the daily Min/Max temperature occurs. By default, it is assumed that the minimum temperature is recorded at 6:00 am, and maximum is at 13:00 pm.

Hour of Day at Daily Min/Max Temp

Hours at min temp: 6
Hours at max temp : 18

Click "**Graph**" button to preview the daily temperature changing with time, as shown in Figure 8.

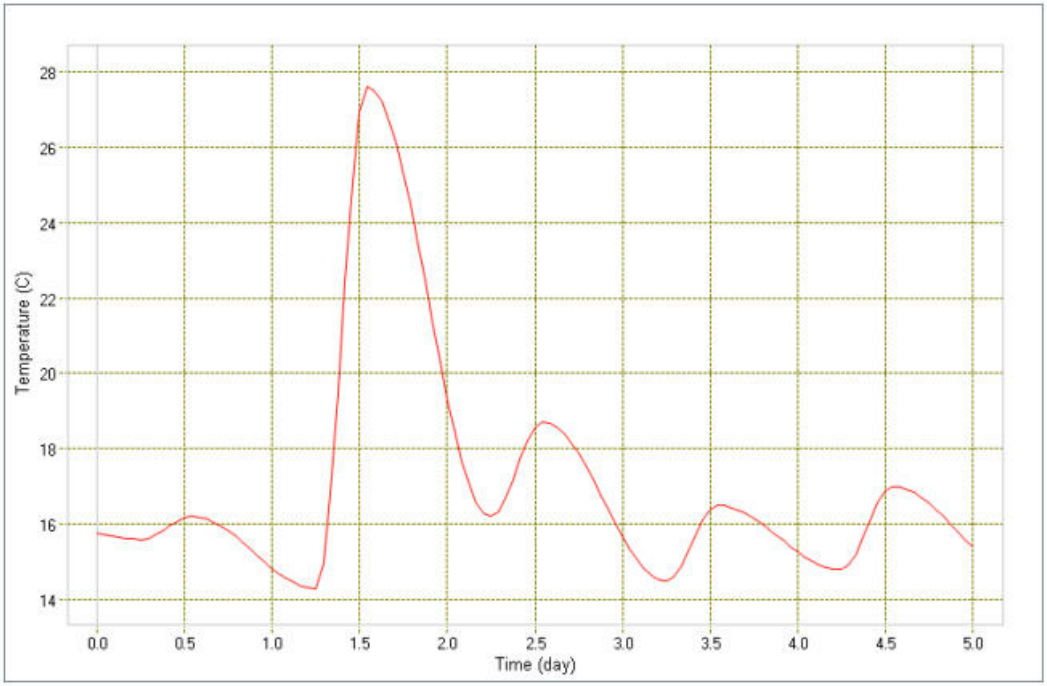


Figure 8 Daily temperature changing time

Click "**OK**" button in the graph,
Click "**OK**" button in the "**Climate Properties**" dialog,
Click "**OK**" button in the "**Climate Manager**" dialog.

3. Modify the end time of model simulation

Considering only 5 daily temperatures are input in the climate temperature, the end time of model simulation is need to change.

Click the menu of **Model** >> **Settings**
Click "**Time**" Tab in the "**Model Settings**" dialog,
Enter the "**End Time**": 5
Click the "Yes" button in the dialog suggesting to update plot specification end time automatically.
Click "**OK**" button.

4. Modified Output settings due to time change

Click the menu of **Model** >> **Reporting** >> **Plot Manager**,

In the **Plots** Tab,
Select all rows in the data grid,
Click "**Multiple Update...**" button,
Click the "**<-**" button to set time steps for temperature output.
Click **OK** button.

In the **Point** Tab,
Select all rows in the data grid,

Click "**Multiple Update...**" button,
Click the "**<-**" button to set time steps for temperature output.
Click **OK** button.

5. Run the model
Click the menu of "**Solve** >> **Analyze**"

6. Analysis of result
See above

Change climate temperature based on hourly temperature

The following steps are to modify model settings so that user can specify the hourly based temperature each day.

1. Save model as a separate model name,
Click the menu of **File** >> **Save As**. A dialog will be opened. In this dialog, do the following:

In General Tab,
Enter the "**New File Name"**: SimpleRoadPavement_1D_hourly_Temp

In Time Tab.
Select "**hr**" from the "**Time Units**" drop-down list. Click **OK** for the dialog suggesting to convert time unit automatically.

Change increment and end time to simulate 1 day
Initial Increment: 0.1
Maximum Increment: 0.5
End Time: 24

Click "**OK"** button.

2. Modified climate temperature settings
Click the menu of **Model** >> **Boundaries** >> **Climate Manager ...** ,
Select the row of "**DemoClimate**" from the data grid,
Double click the (2nd) column of "Applied" in the selected row to open the "Climate Properties" dialog.
Click "**Air Temperature**" Tab
Select "**Data – Spline-Function**" from the "Air Temperature Options: " drop-down list,
Enter the following time and temperature in the data grid.

NOTE:
You can copy dataset from Microsoft Excel and then paste to the data grid in the dialog.

Click the "**Graph...**" button to preview the temperature changing with time.
Click "**OK**" button in the graph dialog.

Time(hr)	Temperature (C)
1	15.85
2	15.8
3	15.75

4	15.7
5	15.65
6	15.6
7	15.65
8	15.7
9	15.75
10	15.8
11	15.85
12	15.9
13	15.95
14	16
15	16.05
16	16.1
17	16.15
18	16.2
19	16.15
20	16.1
21	16.05
22	16
23	15.95
24	15.9

Click **N-Factor** Tab
Change N-Factor to 1, because the temperature at the ground surface is supposed to be equal to the air temperature.

"N-Factor Constant: " 1

Click "**OK**" button in the "**Climate Properties**" dialog.

3. Modified Output settings for the time unit change (**Model** >> **Reporting** >> **Plot Manager**)
Click the menu of **Model** >> **Reporting** >> **Plot Manager**,

In the **Plots** Tab,
Select all rows in the data grid,
Click "**Multiple Update...**" button,
Click the "**<-**" button to set time steps for temperature output.
Click **OK** button.

In the **Point** Tab,
Select all rows in the data grid,

Click "**Multiple Update…**" button,
Click the "**<-**" button to set time steps for temperature output.
Click **OK** button.

4. Run Model
   Click the menu of "**Solve** >> **Analyze**"

5.2 Results and Discussion

a. Display Temperature at Different Layers

1. Click menu of **Window** >> **AcuMesh**,
2. Click menu of **Graphs** >> **Plot Manager**…
3. Click **Point** tab,
4. Select the row with title of "TempAtEachLayer" from the data grid table, double click the selected row. The temperature changing at different location is illustrated in Figure 6.

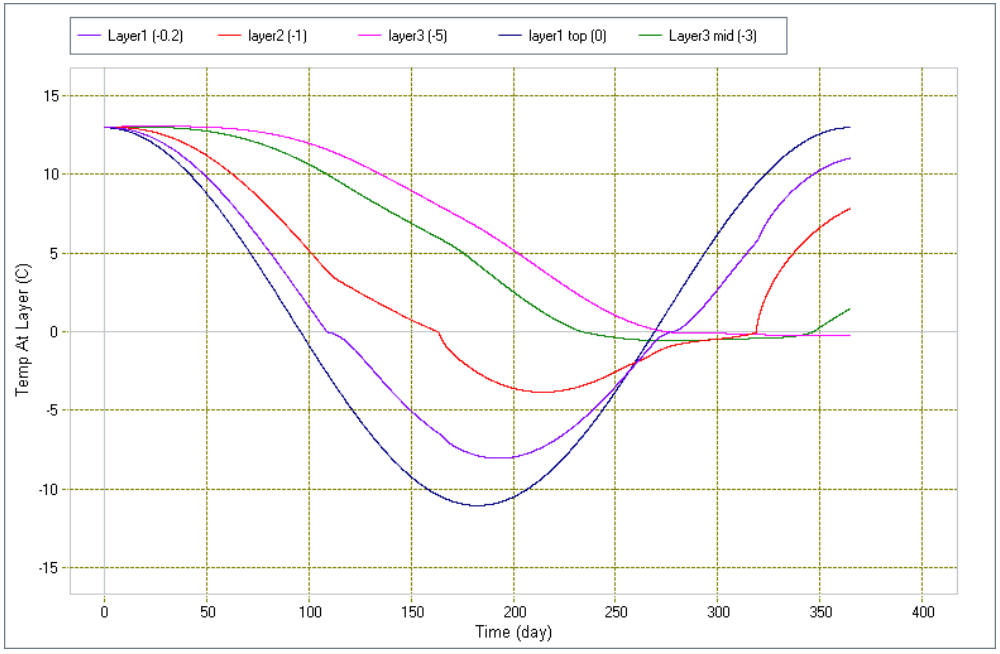


Figure 1. Temperature distribution at different locations.

5. Change Y-axis title into "Temperature (C), as indicating in Figure 1,
6. Copy the graph and paste it to your document by clicking the button indicating in Figure 6.
7. Click **OK** button.

Soil Temperature Distribution along the elevation

1. Click Plots tab,
2. Double click the row with Title of "Temp",
3. The temperature changing along elevation from 0 to -5 m at the time of 180 days is illustrated in Figure 7.

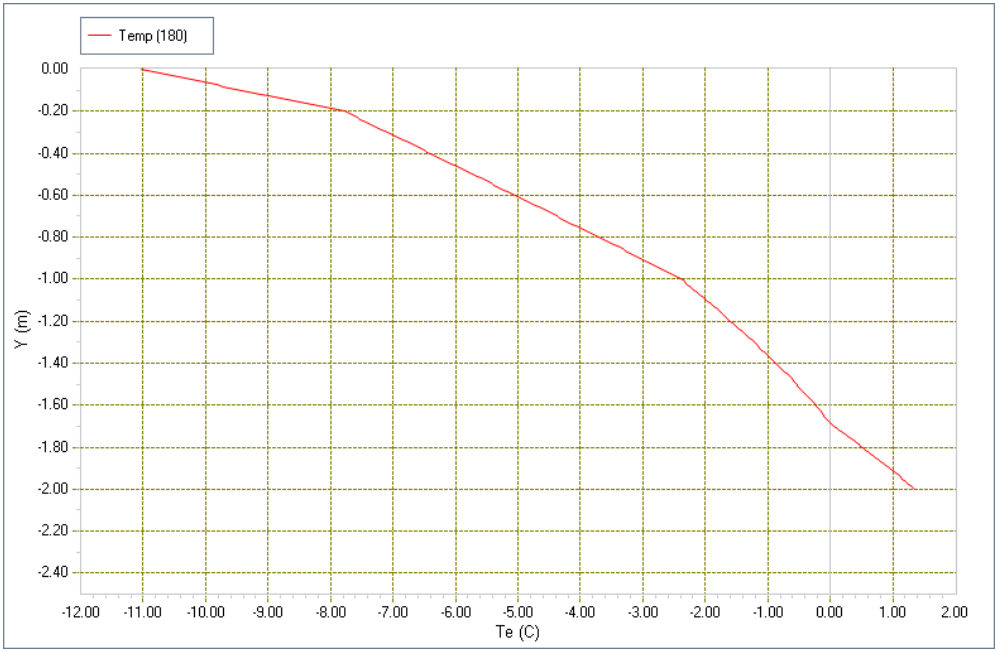


Figure 2 Temperature changing along elevation at the time of day 180

NOTE:

You can use the data file of TempAtEachLayer.txt that is located under the directory of model file running to analyze or compare the simulation result with the measured temperature.